DEVELOP DRAG ESTIMATION ON HYBRID ELECTRIC VEHICLE (HEV) MODEL USING COMPUTATIONAL FLUID DYNAMICS (CFD)

REDZUAN BIN AHMAD

A report submitted in partial fulfillment of the
requirement for the award of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

NOVEMBER 2008

ACKNOWLEDGEMENTS

I would like express my gratefulness to my supervisor, Mr Devarajan A/L Ramasamy for his guidance and help during the whole course of completing my final year project. All his advices and ideas have eventually contributed to the success of this project.

I would like also to acknowledge with much of appreciation to my friends that also supervised by Mr. Devarajan, where is Johari Bin Ismail , Mohd Syazrin Bin Sopnan, Mohd Azrul Bin Mohd Othman , and Shasendran that whose have helped and give co-operation throughout the project and also Instructor Engineer, Mr Mohd Fazli B. Ismail. Then not forgotten to my friend, Hafnizar Bin Muhamad Shaharuddin for allowing me using his car to perform my test experiment.

Last but not least, I would also like to wish my gratitude to all my course mates and other friends for their encouragement, helps, and motivation that inspired me to do well my bachelor degree's final year project. Thank also to all lecturers, technician, and panels presentation that direct and indirectly contributed in completing and improving my quality of project.

ABSTRACT

Develop of drag estimation by using Computational Fluid Dynamics (CFD) on Hybrid Electric Vehicle (HEV) model was carried out on this project. The HEV model here means the Proton Iswara Hatchback body developed by researcher of Automotive Focus Group, Universiti Malaysia Pahang. To develop this HEV model, one of factor needs to consider and studied for giving better efficiency on the road is aerodynamics drag. Therefore, simulation of CFD and FEM have been the key features to aerodynamics drag studies in this project for the HEV model specifically is Proton Iswara Hatcback body. The objectives of this project are to estimate the drag of Proton Iswara Hatcback body at ranging speed between 40km/h to 110 km/h that designed by Computational Aided Design (CAD). The terminology to getting the drag estimation is by using input of CFD then export to FEM analysis to find the value of aerodynamics drag in terms of drag forces and drag coefficient. Besides that, CFD simulation results such as contour and trajectories plot also used to analyze the characteristics of streamlines flow or boundary layer that occurs on the body of this HEV model especially for the forebody, upperbody and rearbody. To achieve these objectives and rationalization made of project, aerodynamics studies, and study of CAD, CFD, and FEA engineering's software needed to optimize the development of aerodynamic design on HEV model and to estimate the drag using CFD and FEA software as an alternative after experimental process. In this project also, a simple experiment was done to validate the CFD simulation analysis. The experiment known as Pressure Experiment that gives valuable results to compare with the simulation results as a validation process to this project.

ABSTRAK

Menentukan aggaran daya rintangan udara terhadap Kenderaan Hibrid Elektrik (HEV) model menggunakan 'Computational Fluid Dynamics' (CFD) menjadi keutamaan dalam projek ini. Kenderaan Hibrid Elektrik model tersebut ialah Proton Iswara Hatchback yang telah di bangunkan oleh sekumpulan penyelidik automotif dikenali sebagai 'Automotive Focus Group' di Universiti Malaysia Pahang. Salah satu faktor yang dipertimbangkan dalam penyelidikan untuk melihat kecekapan HEV model tersebut ialah dari segi daya rintangan aerodinamik. Oleh itu, simulasi menggunakan CFD dan 'Finite Element Model' (FEM) menjadi kunci utama untuk menyelidik dan menganalisa daya rintangan aerodinamik terhadap HEV model dalam projek ini dan secara spesifiknya Proton Iswara Hatchback. Objektif melaksanakan projek ini ialah mendapatkkan nilai daya rintangan udara HEV model pada julat kelajuan, 40km/h hingga 110km/h pada setiap 10km/h. Terminologi mendapatkan nilai daya rintangan udara tersebut ialah menggunakan input daripada CFD seterusnya dihantar kepada FEM untuk dianalisis dan mendapatkan nilai dalam bentuk daya dan kualiti daya rintangan udara. Selain daripada itu, ciri-ciri aerodinamik yang berlaku di bahagian depan, atas dan belakang kenderaan tersebut di analisa menggunakan keratan dan garis aliran plot yang terdapat di dalam perisian CFD. Bagi mencapai objektif projek ini, perisian kejuruteraan seperti CFD, CAD dan FEA perlu di optimumkan untuk mendapatkan anggaran nilai daya dan kualiti daya rintangan udara seterusnya menjadikan CFD dan FEM sebagai alternatif lain selepas proses eksperimen. Dalam projek ini juga, satu eksperimen mudah telah dijalankan untuk mengesahkan keputusan proses simulasi. Eksperimen tersebut dikenali sebagai 'Eksperimen Tekanan' yang memberikan keputusan sangat berguna selepas dibandingkan dengan keputusan simulasi sebagai proses pengesahan terhadap projek ini.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF GRAPHS

LIST OF FIGURES

LIST OF APPENDICES

LIST OF SYMBOLS

D	Drag Force
D_f	Friction Drag Force
L	Lift Force
C_D	Drag Coefficient
Cp	Pressure Coefficient
C_L	Lift Coefficient
ρ	Air Density
A	Frontal Area
b	Length Normal to The Flow
h	Height of the Body
v	Speed of the Body
v_∞	Local Velocity
p_∞	Local Pressure
S_i	Mass-Distributed External Force per Unit Mass
E	Total Energy per Unit Mass
Q_H	Heat Source per Unit Volume
τ_{ik}	Viscous Shear Stress Tensor
q_i	Diffusive Heat Flux

LIST OF ABBREVIATION

HEV	Hybrid Electric Vehicle
CFD	Computational Fluid Dynamic
FEM	Finite Element Model
CAD	Computational Aided Design
CAE	Computational Aided Engineering
RANS	Reynolds-Averaged Navier-Stokes Equation
DNS	Direct Numerical Simulation
3D	Three Dimensional

CHAPTER 1

INTRODUCTION

1.1 Project Background

The importance of aerodynamics to a Hybrid Electric Vehicle (HEV) model needs a development of drag estimation to know how much the car performance on the road against air resistance beside to improve the stability, reducing noise and fuel consumption. In view of the fact that many of car makers like Toyota, Honda and Audi formulate a research and continue develop the HEV model focused on higher propulsion efficiency orderly integrate the energy saving by reduce the rolling resistance of wheel and reduce the drag by aerodynamically losses. At University Malaysia Pahang (UMP), Automotive Focus Group also was developed HEV model to achieve the aim of HEV's control strategy in term of efficiency. Proton Iswara Hatchback body was used by the group to modify the conventional power train to the hybrid power train. As an increasing of drag, the more power of car to do work than reducing the power train efficiency. Therefore, the body of passenger car (Proton Iswara Hatchback) needs to study in term of aerodynamics losses. In aerodynamic field there have two major studies need to be concerned where is study the airflow on the body and estimation of drag. To understand the aerodynamics on the HEV model, flow visualization is the best technique as usual does by wind tunnel. But, in this project Computational Fluid Dynamics (CFD) analysis will be used as the technology of computer simulation to estimate the drag of HEV model after conventional technique due to economical factor.

1.2 Problem Statement

In Universiti Malaysia Pahang, one group of automotive researcher known as Automotive Focus Group were studied the design and development on Proton Iswara Hatchback body as a hybrid electric car. Therefore, the main concerns in aerodynamics fields were focused to study to know the efficiency and performance of that HEV model. The aerodynamics fields consideration in that researches' is drag reduction which is be the most important factor of HEV model design.

Drag will cause many problems on the performance of HEV model like instability, noise and fuel consumption. Thus, in this project the CAD model of Proton Iswara's body was developed to analyze aerodynamics especially on the drag estimation. In addition, using CFD and FEM analysis as a possible procedure were develop the drag estimation and aerodynamics studies on the body due to no wind tunnel in UMP.

1.3 Objectives

1. To estimate the drag of HEV model (Proton Iswara's body)
2. Develop the drag estimation from Computational Fluid Dynamics (CFD) and Finite Element Model (FEM) analysis

1.4 Project Scopes

1. Study of aerodynamics on road vehicle
2. Study of Computer Aided Design (CAD) engineering software
3. Analyze the project with CFD and Finite Element Model (FEM) for various car speeds
4. Optimize the software engineering as a tool to develop aerodynamics design on passenger car.

CHAPTER 2

LITERATURE REVIEW

2.1 Theory of Aerodynamics

At this section, the fundamental of fluids mechanics and basics of aerodynamics were discussed to gain understanding in doing analysis of the project. The basics equation and terms in aerodynamics field or fundamental of fluid mechanics such as Bernoulli's Equation, pressure, lift and drag coefficient, boundary layer, separation flow, and shape dependence were studied.

2.1.1 Bernoulli's Equation

Aerodynamics play main role to defined road vehicle's characteristic like handling, noise, performance and fuel economy [1]. The improvement on the characteristic related through the drag force which is ruled by Bernoulli Equation.
Basic assumptions of Bernoulli's Equation for an air flows are;

1. Viscous effects are assumed negligible
2. The flow is assumed to be steady
3. The flow is assumed to be incompressible
4. The equation is applicable along streamline

$$p + \frac{1}{2}\rho v^2 = constant \qquad (2.1)$$

From equation (2.1) shows the increasing of velocity will case the decrease in static pressure and vise versa. On the movement of road vehicle will produce a distribution velocity that's create the skin friction due to viscous boundary layer which act as tangential forces (shear stress) then contribute drag. Beside that, force due to pressure also created which acts perpendicular to the surface then contribute both lift and drag forces. The Bernoulli's Equation from equation (2.1) gives the important result which is [2], [4], [5];

Static pressure + Dynamic Pressure = Stagnation Pressure.

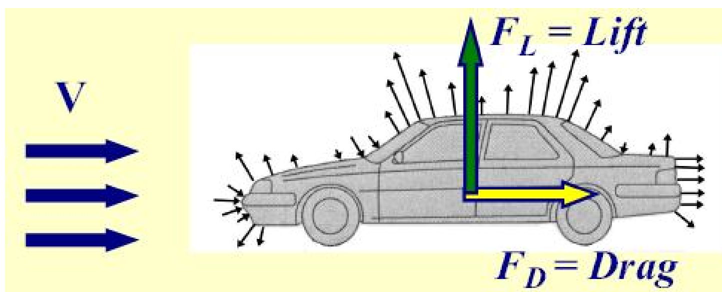

Figure 2.1 Drag and lift force due to pressure from velocity distribution [7]

2.1.2 Pressure, Lift and Drag Coefficient

Drag can generate by two main perspectives [1]:

1. From the vehicles (body)
2. From the moving fluid.

From the two perspectives, three major coefficients were produced from the two basic of aerodynamics forces. The first force is pressure distributions that normal (perpendicular) force to the body which is will produce pressure, drag and lift coefficient. The second force is shear force that tangential (parallel) to the surface of body's vehicle where is contribute drag coefficient only [2], [3].

2.1.2.1 Pressure Coefficient

The equation for coefficient of pressure (*Cp*) due to dynamic pressure can derive as [3],[4] ;

$$Cp = \frac{p - p_\infty}{\frac{1}{2}\rho v_\infty{}^2} \tag{2.2}$$

The equation of dynamic pressure defined as [3],[4];

$$p_{tot} - p_\infty = \frac{\rho}{2} v_\infty{}^2 \tag{2.3}$$

In term of local velocity, the pressure coefficient (only valid for incompressible flow) can derive as [3],[4];

$$Cp = 1 - \frac{v^2}{v_\infty{}^2} \tag{2.4}$$

The form of equation (2.4) is from the relation equation (2.2) and equation (2.5) as shown below [2],[3],[4];

$$p - p_\infty = \frac{1}{2}\rho(v_\infty{}^2 - v^2) \tag{2.5}$$

From the equation (2.4) where the local velocity on velocity is zero, the pressure coefficient is equal to 1.0 and when $v = v_\infty$, the pressure coefficient will be zero. While from equation (2.2) where $p = p_\infty$, *Cp* was become zero also. Pressure coefficient would become negative, since the local velocity is larger than the free stream velocity, v_∞ . Therefore, some typical value of pressure coefficient can summarize on table as shown in Table 2.1 below.

Table 2.1: Typical Values of Pressure Coefficient, *Cp* [3], [4].

Location	**Cp**	**Velocity, v**
Stagnation Point	1.0	0
On body's vehicle	0-1.0	$v < v_\infty$
On body's vehicle	Negative	$v > v_\infty$

2.1.2.2 Drag Coefficient

As was informed before the net drag is produced by both pressure and shear forces, thus the drag coefficient (C_D) for a vehicle body can define as [2], [3], [4];

$$C_D = \frac{D}{\frac{1}{2}\rho v_\infty{}^2 A} \tag{2.6}$$

Where D is the drag and A is the frontal area

Since, the C_D was defined as shown in equation (2.6). Thus, the drag force can derive as;

$$D = \frac{1}{2}\rho v_\infty{}^2 C_D . A \tag{2.7}$$

Besides that, the drag coefficient, C_{df} can derive from friction drag, Df, on a flat plate as [2];

$$C_{d\,f} = \frac{D_f}{\frac{1}{2}\rho v^2 b.l} \tag{2.8}$$

Where D_f is friction drag, b and l are width and length of flat plate

2.1.2.3 Lift Coefficient

The lift force can be determined if the distribution of dynamic pressure and shear force on the entire body are known. Therefore the lift coefficient (C_L) can indicate as [3], [4];

$$C_L = \frac{L}{\frac{1}{2}\rho v_\infty{}^2 A} \tag{2.9}$$

Where L is lift force and A is the frontal area

Pressure and shear stress distribution is difficult to obtain along a surface for non geometry body either experimentally or theoretically but these to value can be obtained by Computational Fluid Dynamics (CFD) [1], [2].

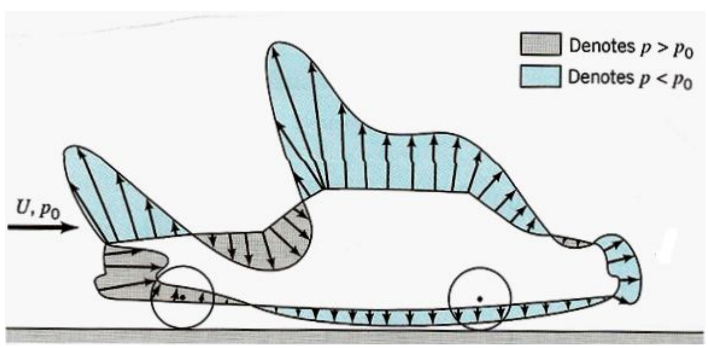

Figure 2.2 Pressure distributions on the surface of an automobile [2]

2.1.3 Boundary Layer

Boundary layer study in aerodynamics can be describe on a flat plate where is develop with two types flow which is laminar and turbulent flow. Due to fluid viscosity, a thin layer will exist when the velocity parallel to the static flat plate and then gradually increase the outer velocity. The thickness of boundary layer also increases with the distance along the flat plate's surface [2], [3], [4].

Normally, the boundary layer is start from laminar flow and develops into turbulent flow. These two types of flow can determined with change of Reynolds number. Between the laminar and turbulent, form of transition region start occur when the change on laminar flow into turbulent flow [2], [3], [4]. The variation of boundary layer thickness can be seen in Figure 2.3;

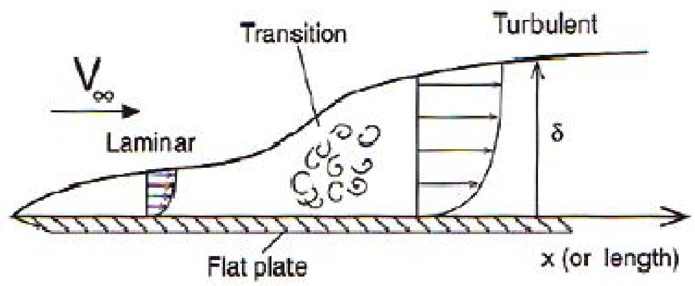

Figure 2.3 Variation of boundary layer thickness along flat plate [3]

2.1.4 Separation Flow

Separation flow can define as the fluid flow against the increasing pressure as far as it can; at point the boundary layer separates from the surface where the fluid within the boundary layer does not have such an energy supply [2].

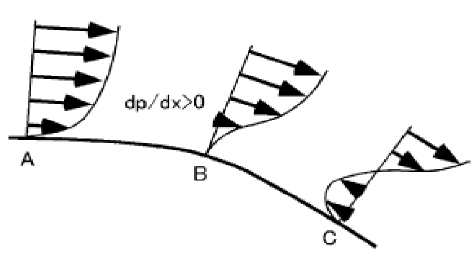

Figure 2.4 Schematic of velocity profile around a rear end [6]

In automobile shape, the rear end of vehicle becomes increasingly lower as the flow moves downstream then the extended airflow was formed there. Thus, it causes the downstream pressure increase while creates reverse force acting alongside the main flow and generates the reverse flow at downstream at point C as showed in Figure 2.4.

At Point A, no reverse occur because the momentum of the boundary layer is widespread over the pressure gradient. Between the Point A and Point C, the momentum of boundary layer and pressure gradient are balanced as stated at separation on Point B. The reverse force acting on separation point C is due to the viscosity of air (losses of momentum as it moves downstream) [6].

2.1.5 Shape Dependence

As discussed before the drag was depend on the shape of vehicle [1]. The Drag coefficient from equation (2.6) and equation (2.8) shown clearly the frontal area is give effect on the drag and lift coefficient means that increasing of frontal area or more blunt of body shaped will increase the both coefficient orderly increase the drag.

For the case, $0<Cd<1$ and $v< v_\infty$, the drag, $D=½\ \rho v_\infty^2 Cd.A$ that was defined from equation (2.7) [2]. Thus from the definition we can conclude that area of frontal area projected of composite body as [2], [8];

$$A=b.h \tag{2.10}$$

Where b is the length normal to the flow, and h is the height of the body